March 10, 2006

Study of Residual Winding Tension in Focusing Solenoid

Bob Wands

Introduction and Summary

The test solenoid is wound with 20 layers of 0.808mm diameter NbTi superconducting wire. Each layers contains 118 turns, for a total of 2356 turns in the solenoid.

Winding tension is 20 N, producing a winding stress of 39 MPa in the wire. As winding progresses, the stainless steel and copper spool on which winding is done progressively compresses, building up compressive hoop stress in response to the tensile hoop stress of the wire.

However, the winding of one layer will act to reduce the tension of the layers underneath; the intent of the winding strategy is to wind with a tension sufficient to allow the innermost layer to maintain a slight amount of tensile hoop stress.

This study indicates that the innermost layer will retain approximately 3 MPa of tensile hoop stress after winding is completed.

Material Properties

The material properties used in this analysis area given in Table I.

Table I. Material Properties

Material	Young's Modulus - GPa
Stainless Steel	199
Copper	70
NbTi	97.9

The Finite Element Model

An axisymmetric ANSYS finite element model was created using the ANSYS plane42 structural four-node quadrilateral element. The degrees of freedom are displacements in the radial and axial direction. This model is capable of simulating the solenoid with any number of layers completely wound. The physical thickness of a coil layer was adjusted to give the correct cross sectional area of 118 turns of 0.808mm wire, and hence the correct layer stiffness; this thickness is 0.6 mm. Layer-to-layer interfaces were simulated with gap elements, which provide resistance to compression only.

The finite element model is shown in Fig. 1.

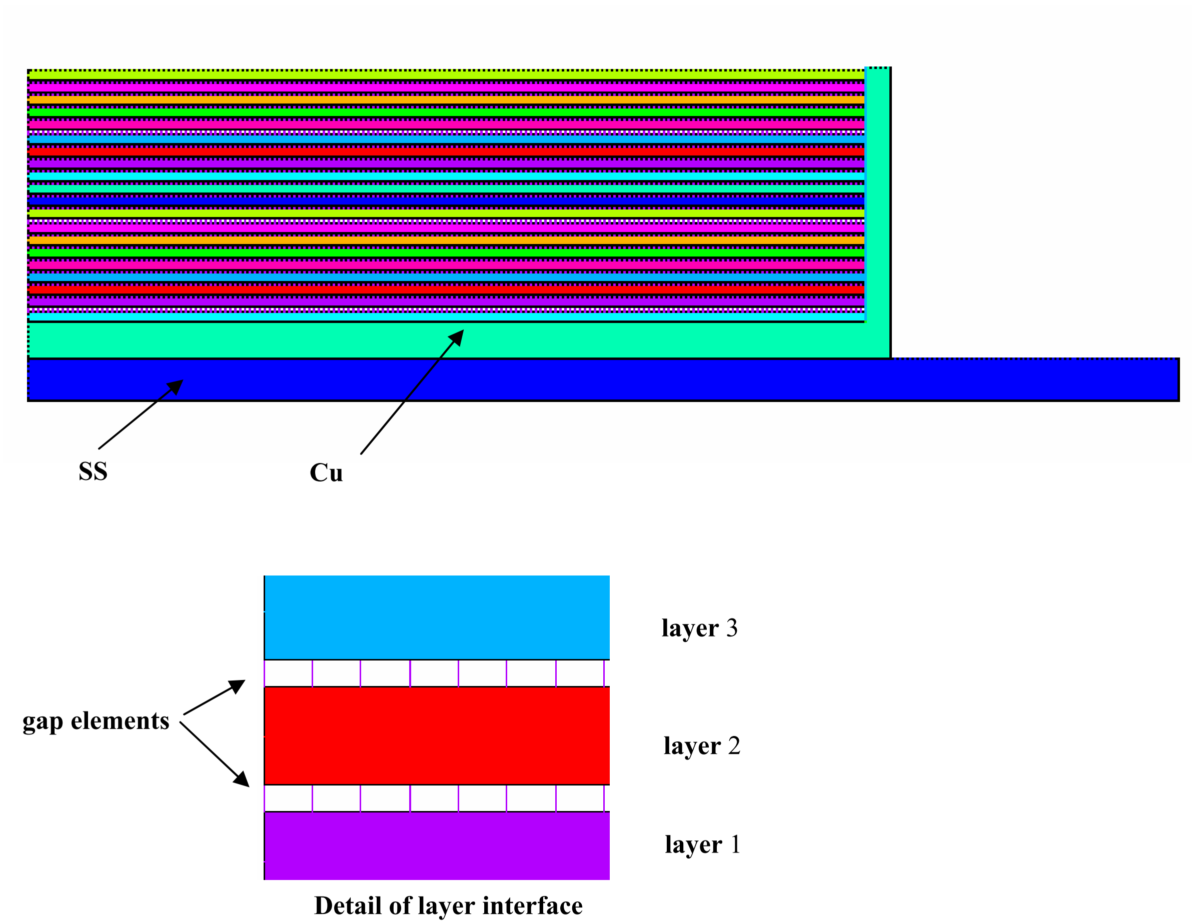


Figure 1. Finite Element Model for Winding Study

Simulation of Winding – ANSYS Optimization

The hoop tension of each completely wound layer is generated in the finite element model by thermal strains, produced by specifying an artificial temperature difference (relative to a reference temperature of 293 K) at each element in the layer. The spool is stiffened at its end by a copper flange, and a uniform temperature difference will tend to create larger hoop stresses at the ends of the solenoid than at its center. This is due to the stiffening effect of the end flange. Therefore, the optimization module of ANSYS was used to find a distribution of temperature difference for each layer which produces an average stress of 39 MPa, with a variation of no more than +/- 1.6 MPa.

The assumed distribution of ΔT over a typical winding is described in Fig. 2 below. A constant temperature difference is applied over the length D; between D and L, the temperature difference is adjusted downward as the element location, d, moves toward the end of the solenoid. ANSYS optimized each of the parameters A, B, D, and ΔT_0 to give the desired average stress and variation.

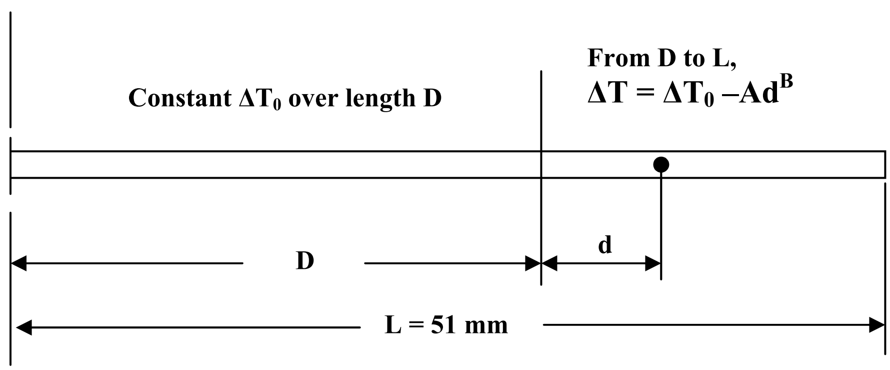


Figure 2. Strategy for Adjusting ΔT along Length of Layer

A zero value of thermal contraction coefficient was applied in the axial and radial directions, so that they would not participate in the analysis. A constant hoop direction thermal contraction coefficient of 1e-4 mm/mm-K was applied to the winding. This is a non-physical value used only for the purposes of generating winding hoop stress. This value gives typical temperature differences of between 3 and 5 degrees (element temperatures of between 286 and 289 K) to achieve the target layer tension.

This method of applying winding tension requires that the artificial thermal contractions be taken into consideration in the finite element simulation of cooldown. That accounting will be described in the report of that analysis.

Results

Following the initial optimization runs, which were used to establish the artificial thermal contractions necessary to give the required winding tension, the model was run 20 times, each time adding one layer of wire.

Fig. 4 shows the progressive build-up of compressive hoop stress at the inner radius of the stainless steel portion of the spool (line A-B of Fig. 3) as the solenoid is wound. The absolute value of this stress is a maximum toward the center of the spool, and drops off toward the end, due to the stiffening effect of the flange at the spool end.

Fig. 5 shows the same build-up, this time calculated at the inner radius of the copper portion of the spool (line C-D of Fig. 3).

The maximum hoop stress as a function of layers completed is shown in Figs 6 and 7 for the stainless steel and copper portions of the spool, respectively.

The tension in each layer as progressive layers are wound is shown in Fig. 8. All windings begin with a tension of approximately 39 MPa; the innermost layer drops in tension as each successive layer is added. However, as the figure shows, there is still a small amount of tension (about 3 MPa) left in the innermost layer after the twentieth layer is wound.

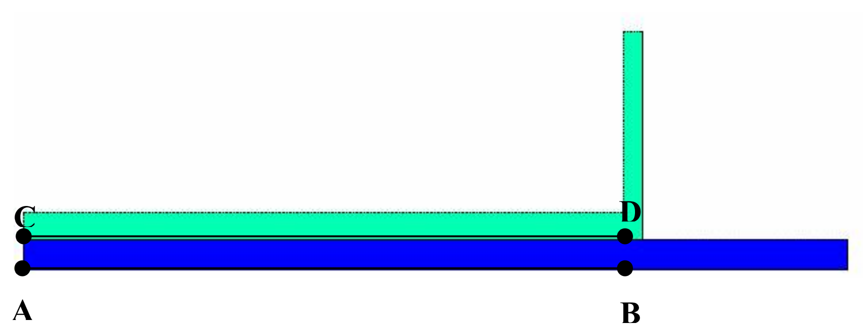


Figure 3. Lines used for Path Plots in Figs. 4 and 5

Figure 4.

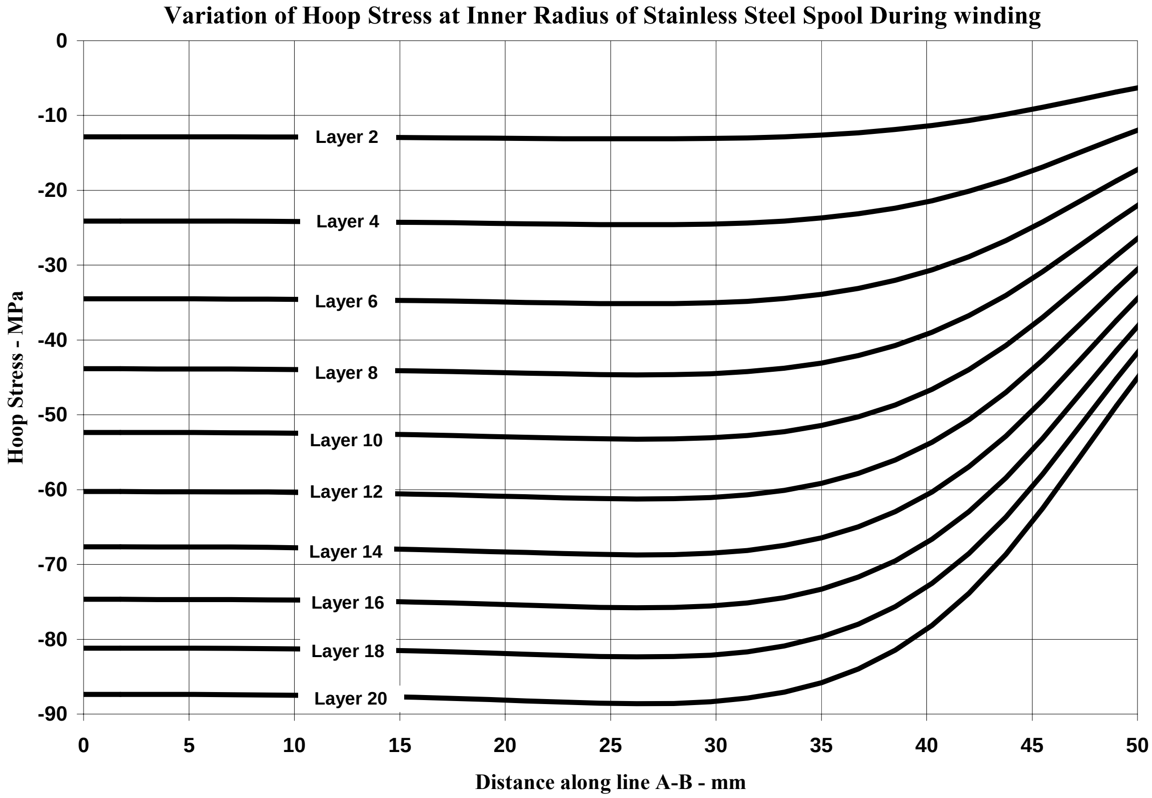


Figure 5.

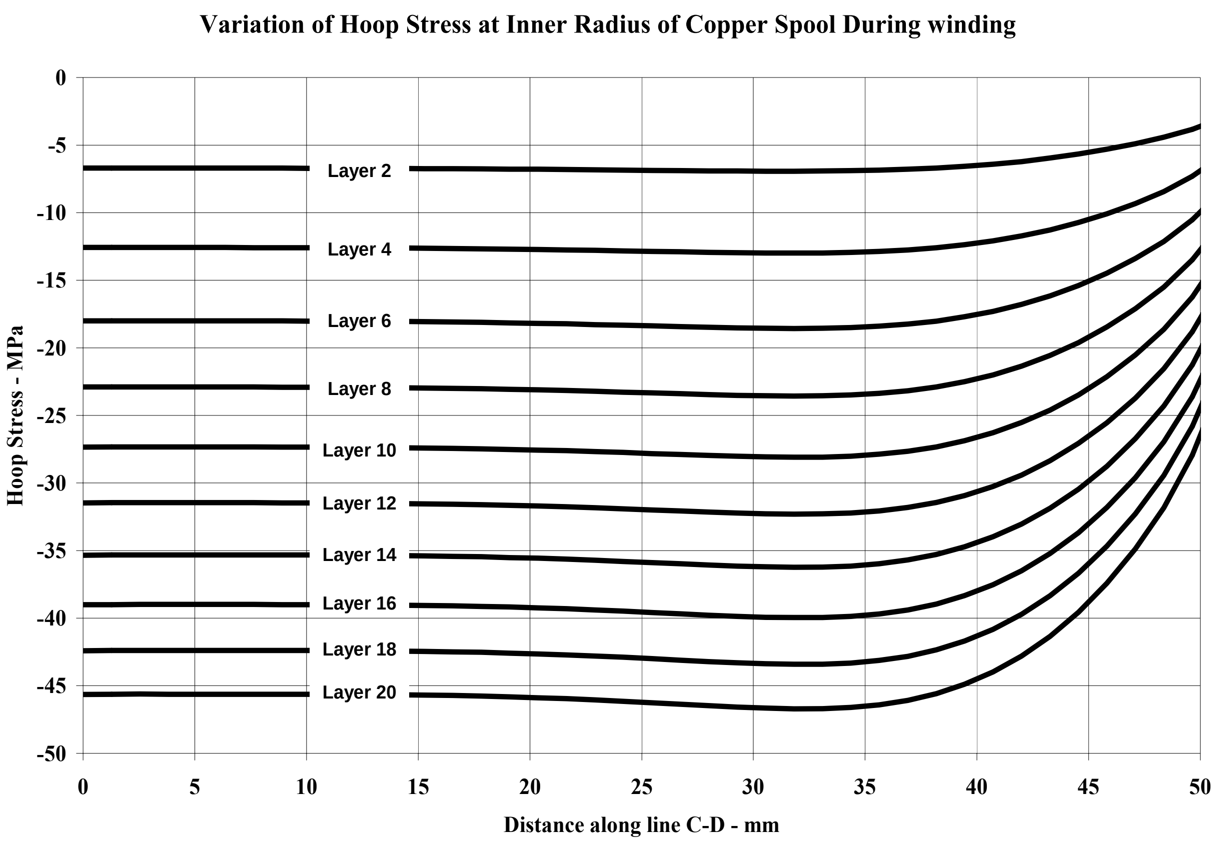


Figure 6.

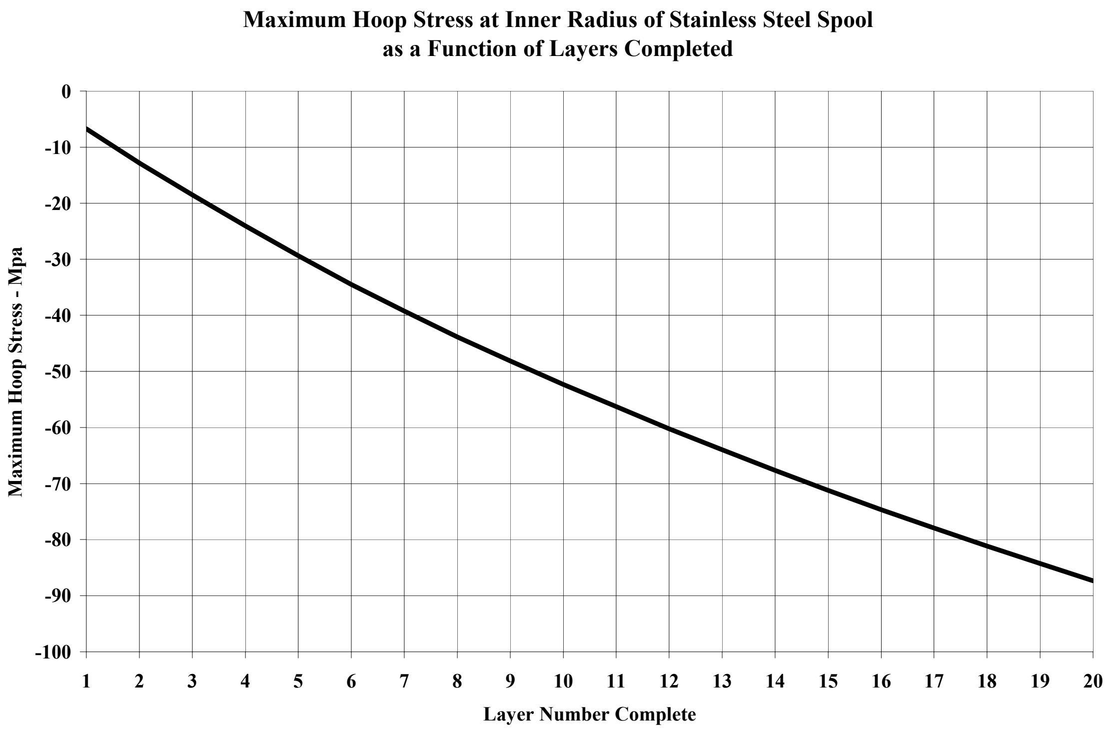


Figure 7.

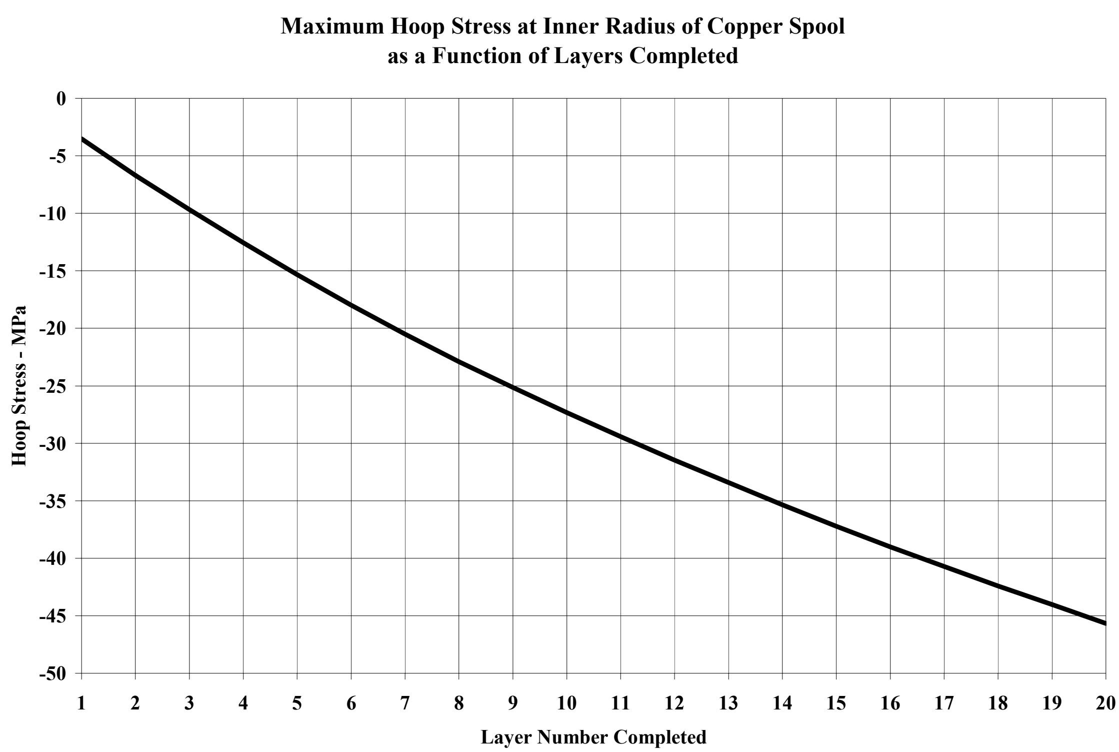
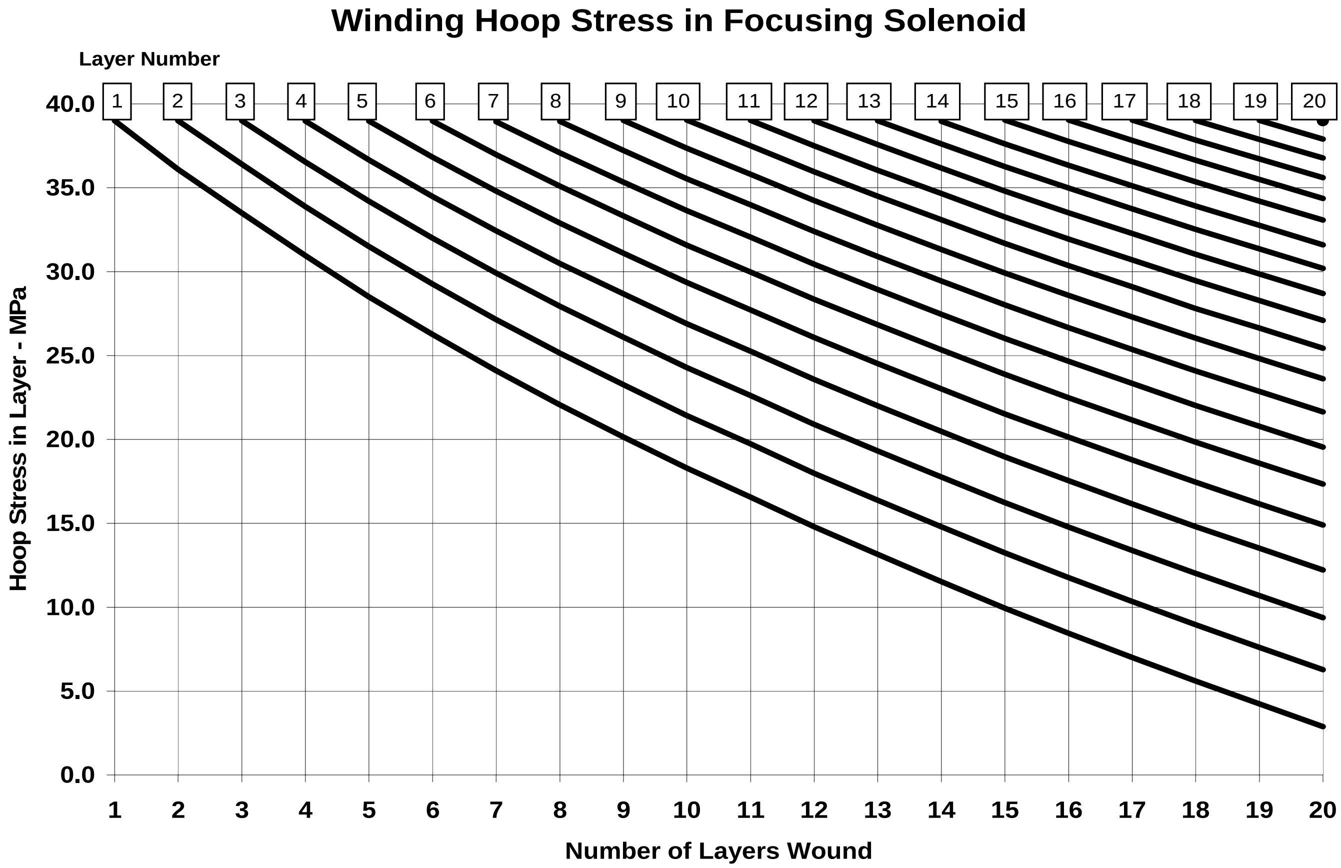


Figure 8.



Conclusion

The winding tension is sufficient to compress the spool substantially while still maintaining a small amount of tension on the first layer of the winding. During cooldown, the spool is further compressed due to the greater thermal contraction of the coil compared to the copper and stainless steel components of the spool. This places the entire system in a favorably prestressed state at the beginning of energizing, a state that should ensure that, under Lorentz forces, layers do not separate from each other, or from the spool surface.